4.1 Why do We Carry Out Meshing?

The basic idea of FEA is to make calculations at only limited (Finite) number of points and then interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it’s just not possible to solve the problem in this format. Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).
### 4.2 Types of Elements

<table>
<thead>
<tr>
<th>Elements</th>
<th>1-D</th>
<th>2-D</th>
<th>3-D</th>
<th>Other</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-D</td>
<td><img src="image1" alt="1-D Element Diagram" /></td>
<td><img src="image2" alt="2-D Element Diagram" /></td>
<td><img src="image3" alt="3-D Element Diagram" /></td>
<td><img src="image4" alt="Other Element Diagram" /></td>
</tr>
</tbody>
</table>

#### 1-D Elements
- **Shape**: Line
- **Additional Data**: Remaining two dimensions (e.g., area of cross section)
- **Type**: Rod, bar, beam, pipe, axisymmetric shell, etc.
- **Applications**: Long shafts, beams, pin joints, connection elements, etc.

#### 2-D Elements
- **Shape**: Quad, tria
- **Additional Data**: Remaining dimension (e.g., thickness)
- **Type**: Thin shell, plate, membrane, plane stress, plane strain, axisymmetric solid, etc.
- **Applications**: Sheet metal parts, plastic components like an instrument panel, etc.

#### 3-D Elements
- **Shape**: Tetra, penta, hex, pyramid
- **Additional Data**: None
- **Type**: Solid
- **Applications**: Transmission casing, engine block, crankshaft, etc.

#### Other Elements
- **Type**: Mass, Point element, concentrated mass at the center of gravity of the component
- **Type**: Spring, Translational and rotational stiffness
- **Type**: Damper, Damping coefficient
- **Type**: Gap, Gap distance, stiffness, friction
- **Type**: Rigid, RBE2, RBE3
- **Type**: Weld

#### Diagrams
- ![1-D Element Diagram](image1)
- ![2-D Element Diagram](image2)
- ![3-D Element Diagram](image3)
- ![Other Element Diagram](image4)

### 4.3 How to Decide the Element Type

**Element type selection**

- **Geometry size and shape**
- **Type of analysis**
- **Time allotted for project**

#### a. Geometry size and shape

For an analysis, the software needs all three dimensions defined. It cannot make calculations unless the geometry is defined completely (by meshing using nodes and elements).

The geometry can be categorized as 1-D, 2-D, or 3-D based on the dominant dimensions and then the type of element is selected accordingly.

#### 1-D element
- Used for geometries having one of the dimensions that is very large in comparison to the other two.
The shape of the 1-D element is a line. When the element is created by connecting two nodes, the software knows about only one out of the 3 dimensions. The remaining two dimensions, the area of the cross section, must be defined by the user as additional input data and assigned to the respective elements.

**Practical Example:** Long shaft, rod, beam, column, spot welding, bolted joints, pin joints, bearing modeling, etc.

**2-D Element:** Used when two of the dimensions are very large in comparison to the third one.

2-D meshing is carried out on a mid surface of the part. 2-D elements are planar, just like paper. By creating 2-D elements, the software knows 2 out of the 3 required dimensions. The third dimension, thickness, has to provided by the user as an additional input data.

**Why is 2-D meshing carried out on a mid surface?**

Mathematically, the element thickness specified by the user is assigned half on the element top and half on the bottom side. Hence, in order to represent the geometry appropriately, it is necessary to extract the mid surface and then mesh on the mid surface.

**Practical Examples:** All sheet metal parts, plastic components like instrument panels, etc. In general, 2-D meshing is used for parts having a width / thickness ratio > 20.
Limitations of mid surface and 2-D meshing –
2-D meshing would lead to a higher approximation if used for
- variable part thickness
- surfaces are not planner and have different features on two sides.

3-D Element: used when all the three dimension are comparable

Practical Examples: Transmission casing, clutch housing, engine block, connecting rod, crank shaft etc.

b. Based on the type of analysis:
Structural and fatigue analysis - Quad, hex elements are preferred over trias, tetras and pentas.
Crash and Non-linear analysis – Priority to mesh flow lines and brick elements over tetrahedron.
Mold flow analysis – Triangular element are preferred over quadrilateral.
Dynamic analysis – When the geometry is borderline between the classification of 2-D and 3-D geometry, 2-D shell elements are preferred over 3-D. This is because shell elements being less stiffer captures the mode shapes accurately and with a fewer number of nodes and elements.

c. Time allotted for project:
When time is not a constraint, the appropriate selection of elements, mesh flow lines, and a good mesh quality is recommended. Sometimes due to a very tight deadline, the analyst is forced to submit the report quickly. For such situations
1) Automatic or batch meshing tools could be used instead of time consuming but structured and good quality providing methods.

2) For 3-D meshing tetras are preferred over hexas.

3) If the assembly of several components is involved then only the critical parts are meshed appropriately. Other parts are either coarse meshed or represented approximately by 1-D beams, springs, concentrated mass, etc.

4.4 Can We Solve Same Problem Using 1-D, 2-D and 3-D Elements

Is it not possible to use 3-D elements for long slender beams (1-D geometry), for sheet metal parts (2-D geometry), and 2-D shell elements for representing big casting parts?

The same geometry could be modelled using 1-D, 2-D, or 3-D elements. What matters is the number of elements and nodes (DOF), the accuracy of the results, and the time consumed in the analysis.

For example, consider a cantilever beam with a dimension of 250 x 20 x 5 mm that is subjected to a 35 N force:

1-D beam model

\[ N = 2 \quad E = 1 \]

Total DOF = 6 x 2 = 12

2-D Shell Mesh

\[ N = 909 \quad E = 800 \]

Total dof = 909 x 6 = 5454
3-D Tetra Mesh

\[ N = 17,448 \quad E = 9,569 \]

Total dof = 17,448 x 3 = 52,344

<table>
<thead>
<tr>
<th></th>
<th>Nodes</th>
<th>Elements</th>
<th>Stress N/mm²</th>
<th>Displacement mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>--</td>
<td>--</td>
<td>105</td>
<td>4.23</td>
</tr>
<tr>
<td>1-D</td>
<td>2</td>
<td>1</td>
<td>105</td>
<td>4.23</td>
</tr>
<tr>
<td>2-D</td>
<td>909</td>
<td>800</td>
<td>103</td>
<td>4.21</td>
</tr>
<tr>
<td>3-D</td>
<td>17,448</td>
<td>9,569</td>
<td>104</td>
<td>4.21</td>
</tr>
</tbody>
</table>

4.5 How to Decide Element Length

1) Based on previous experience with a similar type of problem (successful correlation with experimental results).

2) Type of analysis: Linear static analysis could be easily carried out quickly with a large number of nodes and elements, but crash, non-linear, CFD, or dynamic analysis takes a lot of time. Keeping control on the number of nodes and elements is necessary.

3) Hardware configuration and graphics card capacity of the available computer. An experienced CAE Engineer knows the limit of the nodes that can be satisfactorily handled with the given hardware configuration.

Suppose you are a part of a newly formed CAE group (no clear guidelines are available, and there is no experienced person in the group): In the first run, accept the default element length. Mesh with the basic rules of thumb discussed in this book. Then run the analysis and observe the high stress regions. Remesh the localized areas of high stress (with smaller element length) and solve again. Compare the difference in the original and the new results. Continue the process until convergence is achieved (5 to 10% difference in strain energy / maximum stress value).

4.6 How to Start Meshing

1) Spend a sufficient amount of time studying the geometry

A common observation is that CAE engineers start meshing immediately, without properly understanding the geometry and paying attention to all of the requirements and instructions provided. Observing the geometry several times and thinking about it from all angles is strongly suggested. Mental visualization of the steps is the first step in the right direction of creating a good meshing.

2) Time estimation:
Now a days the trend is towards the client or boss specifying the estimated time for a given job to the service provider or subordinate. Sometimes it is decided based on a mutual understanding. A time estimation is very relative and one can find a lot of differences in estimation by different engineers (as much as 2 to 3 times). Usually a less experienced person will estimate more time. Also if someone is handling the job for the first time, then he/she will require more time. If similar kinds of jobs are given to the same engineer again and again, the meshing time would reduce drastically.

3) Geometry check:

Generally CAD data is provided in *.igs format. Geometry cleanup is an integral part of the meshing activity. CAE engineers should at least have the basic knowledge of CAD. Before starting the job, the geometry should be carefully checked for:

- Free edges
- Scar lines
- Duplicate surfaces
- Small fillets
- Small holes
- Beads
- Intersection of parts (assembly of components)

If suppressing fillets, small holes, beads, or the generation of a mid surface is required for meshing, then why isn’t the CAD data provided in the way needed for CAE by the CAD engineers?

Yes, theoretically that would be an ideal situation, but practically everyone works with a very tight schedule and target dates. CAD data is generated keeping in mind the final drawing to be released for manufacturing. The same CAD model is provided simultaneously to the tools and jig /fixture manufactures, vendors, purchase engineers, and CAE engineers, etc.

The simplification required for a FEA is understood better by a CAE engineer than a CAD engineer. All meshing software provides special tools for geometry cleanup and simplification, which are usually much faster than CAD software. Many times, for complicated geometry, surfacing operations fail in CAD software and it could be easily handled by the CAE engineer by avoiding the geometry and generating the mesh using manual or special meshing operations.

4) Symmetry check:

- Complete part symmetry

Meshing only a quarter of the plate and reflecting it twice is advisable.
Sub part symmetry, repetition of features, and the copy/paste command

Meshing the highlighted 22.5° portion and then using reflection and rotation would lead to a faster mesh as well as the same structure of elements and nodes around the critical areas (holes).

5) Selection of type of elements:
In real life, we rarely use only one type of element. It is usually a combination of different types of elements (1-D, 2-D, 3-D, and others).

In the above figure, the handle of the bucket is modelled by beam (1-D) elements, the bucket body uses shell (2-D) elements, and the connection between the handle and the bucket body through RBE2 (rigid) elements.

6) Type of meshing:
   i. Geometry based – The mesh is associated to the geometry. If the geometry is modified, the mesh will also get updated accordingly (automatically). The boundary conditions could be applied on the geometry like a surface or edge, etc.
ii. **FE based** – Mesh is non associative. The boundary conditions are applied on the elements and nodes only.

7) **Joint modeling**:
   a. Special instructions for bolted joints (specific construction around holes)
   b. Spot and arc weld
   c. Contact or gap elements and the requirement of the same pattern on 2 surfaces in the contact
   d. Adhesive joint

8) **Splitting the job**:
When there is little time or when engineers in other group are sitting idle, then the job could be split among several engineers by providing a common mesh on the interfaces.

### 4.7 Meshing Techniques

<table>
<thead>
<tr>
<th></th>
<th>Automatic / Batch</th>
<th>Mapped (or Interactive)</th>
<th>Manual (Special commands: Spline, Ruled, Drag / extrude, Spin / rotate etc.)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Time required for meshing</strong></td>
<td>↓</td>
<td>= (intermediate i.e. more than auto but less than manual)</td>
<td>↑</td>
</tr>
<tr>
<td><strong>Geometry required</strong></td>
<td>✓</td>
<td>✓</td>
<td>X</td>
</tr>
<tr>
<td><strong>No. of nodes and elements generated</strong></td>
<td>↑</td>
<td>=</td>
<td>↓</td>
</tr>
<tr>
<td><strong>User friendliness</strong></td>
<td>↑</td>
<td>=</td>
<td>↓</td>
</tr>
<tr>
<td><strong>User’s control over the mesh</strong></td>
<td>↓</td>
<td>=</td>
<td>↑</td>
</tr>
<tr>
<td><strong>Structural mesh (flow lines)</strong></td>
<td>↓</td>
<td>=</td>
<td>↑</td>
</tr>
<tr>
<td><strong>Experience or skill required</strong></td>
<td>↓</td>
<td>=</td>
<td>↑</td>
</tr>
<tr>
<td><strong>Patience</strong></td>
<td>↓</td>
<td>=</td>
<td>↑ (specially for brick / hex)</td>
</tr>
</tbody>
</table>

Batch meshing / Mesh adviser – Now a days, all software provide special programs for automatic geometry clean up and meshing with little or no interaction from the user. The user has to specify all the parameters like minimum hole diameter, minimum fillet radius, average and minimum element length, quality parameters, etc. and the software will run a program to produce the best possible mesh by fulfilling all or most of the specified instructions. Though these programs are still in the initial stage and for many applications the output is not acceptable, the research is in progress and its performance will surely improve in the coming years.
4.8 Meshing in Critical Areas

Critical areas are locations where high stress locations will occur. Dense meshing and structured mesh (no trias / pentas) is recommended in these regions. Areas away from the critical area are general areas. Geometry simplification and coarse mesh in general areas are recommended (to reduce the total DOFs and solution time).

How would I know about the critical areas before carrying out an analysis?

After going through a previous analysis of a similar part (carried out by your colleague or a senior in the group) one can get a fairly good idea about the probable locations of the high stress. But suppose there is no past record and you are doing it for the first time, then run the analysis with a reasonable element length and observe the results. High stress regions are critical and could be remeshed with a smaller element length in the second run.
Rules for modeling holes and fillets:

<table>
<thead>
<tr>
<th>Critical area</th>
<th>General area</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Circle" /></td>
<td><img src="image2.png" alt="Square" /> <img src="image3.png" alt="Hexagon" /></td>
</tr>
<tr>
<td>Minimum 12 elements around the hole</td>
<td>4 to 6 elements</td>
</tr>
<tr>
<td><img src="image4.png" alt="Curved Edge" /></td>
<td><img src="image5.png" alt="Curved Edge" /></td>
</tr>
<tr>
<td>Minimum 3 elements on fillet.</td>
<td>Suppress small fillets, 1 element for large fillets.</td>
</tr>
</tbody>
</table>

Mesh transition techniques and flow lines:

- ![Triangle](image6.png) ![Triangle](image7.png) 1 to 3
- ![Square](image8.png) ![Square](image9.png) 1 to 3
- ![Rectangle](image10.png) ![Rectangle](image11.png) 2 to 4
- ![Rectangle](image12.png) ![Rectangle](image13.png) 2 to 4
- ![Rectangle](image14.png) ![Rectangle](image15.png) 1 to 2
- ![Rectangle](image16.png) ![Rectangle](image17.png) 1 to 2 x 2
4.9 Mesh Display Options

1. Shell Mesh

  a. *Mixed mode: Geometry – line, Mesh – solid*

This is the most common and preferred way of working.

  b. *Line mode: Geometry and Mesh – both line*

This mode is preferred for brick meshing, for internal mesh adjustment / modifications.

  c. *Solid: Geometry and Mesh – both solid*
This mode is not preferred for regular meshing but is very useful after the completion of the job. It helps to check the mesh deviation from the geometry and to find the kinks or abrupt changes in the mesh.

2. Brick Mesh

a. Line mode options

The figure on the left is used for viewing the internal details while the figure on the right is used for checking free faces inside the mesh.

- Nipple Cream
b. Solid mode

The solid view is commonly used during regular meshing. The shaded view is used for checking the kinks or deviation of the mesh from geometry and the shrink view is used for checking for free faces and for missing or extra 1-D elements on the edge of the element.

4.10 Understanding Element Behavior

To successfully complete a finite element analysis, you must understand the behavior of various types of elements. A deep theoretical knowledge of element formulations is not necessarily required although a fundamental knowledge of how each element type behaves is essential in the selection of the appropriate element type(s) which will lead to proper interaction with applied loads and boundary conditions.

Finite element models consisting of a single element are one method of studying the mechanics of elements. The inputs and outputs can be studied in detail and compared to solid mechanics solutions. This method is useful for understanding the sign and naming conventions used by a particular solver.

The following diagrams depict single element models, each with several load cases applied in conjunction with a minimum set of boundary conditions. Adding more than the required boundary conditions can be used to learn even more about element behavior.

DOFs are important because they dictate the ability of the elements to model a given problem and also dictate whether or not elements are compatible with each other. Further discussion on element compatibility will follow.
**Rod Element**

Example of *rod* element

Nodes: 2 nodes
DOFs: 3 or 6 degrees of freedom per node

**Beam Element**

Example of *Beam* element

Nodes: 2 nodes
DOFs: 6 degrees of freedom per node
Shell Element

First Order  4 or 3  nodes
Second Order 6 or 8  nodes
DOFs  6  degrees of freedom per node

Example of Shell elements (CTRIA3, CQUAD4, CTRIA6 AND CQUAD8)

Solid Element

First Order  4, 5, 6, 8  nodes
Second Order 8, 12, 15, 20  nodes
DOFs  3  degrees of freedom per node

Tetrahedron, Pyramid, Penta and Hexa elements
Higher Order Elements

Higher order elements are those with one or more mid-side nodes, or geometry based elements, such as p-version elements. These types of elements offer the benefits of ease of modeling and a higher degree of accuracy per element. P-version type elements also have a built-in ability to check convergence by increasing the integration level although it more difficult to understand their fundamental behavior.

Higher order elements give rise to issues such as the sophisticated methods required to apply pressure to the face of a shell element. The required distribution of nodal loads to accomplish the same resultant force ($F = P \times A$) on a 4-node and an 8-node shell element is shown below.

Consistent Pressure Loads for Shells ($F = P \times A$)

Most codes handle these details, but you should understand these and the other fundamentals of higher order elements to avoid confusion. Higher order elements are most often used in 3-D solid modeling because the potential to reduce modeling effort and the number of elements required to capture the geometry is greater. Solution time is not often reduced however because the global stiffness matrix is based on nodal DOF in the model.

Plane Stress and Strain Element

Plane stress: $\sigma_z = \tau_y = \tau_z = 0 \, \varepsilon_z \neq 0$

A thin planar structure with constant thickness and loading within the plane of the structure for example:
**Plane strain:**

\[ \varepsilon_z = \gamma_y = \gamma_x = 0 \quad \sigma_y \neq 0 \]

A long structure with a uniform cross section and transverse loading along its length, for example:

![Plane strain diagram](image)

### 4.11 Element Selection

Element selection is based on the type of problem you want to run, boundary conditions, geometry considerations, and results required. Most problems can be solved many different ways and there is no "right" answer to the question of element selection, but making a good choice can reduce effort, computer time, and errors in the results. Often the solver you choose to solve the problem will have limitations for some element types and not for others restricting element selection.

**Masses (0-D elements)**

Masses are point load masses that are generally used to represent attached structures at their centroids. This is an extremely good way to represent otherwise complex structures when the detailed is not required.

**Beams (1-D elements)**

Beams are characterized by long and slender members, such as a space frame or a formula racing suspension. Bridge members are also good examples of beams or spars. Some examples of 1-D elements are listed below:

- Rods
- Spars
- Beams
- Welds
- Rigid

Beams are very useful because of the flexibility in modeling complex cross-sections without modeling the geometry, but the burden of maintaining the detailed information is upon the user. In addition very accurate stress and deflection results are achievable with beam elements, but the visualization of the results is sometimes difficult. Rods and Spars are essentially 2-D beams and are great for in plane problems. Welds and rigid elements are used for defining constraint equations between nodes. Generally this results in an independent node and a dependent node(s) that form a set of equations that are placed in the stiffness matrix.
Plates (2-D element)

Plates are 2-D elements that represent 3-D space by assuming an infinite depth, fixed depth, or axisymmetric geometry. They have a reduced stiffness matrix and therefore reduced solution time with no loss in accuracy if the assumptions for the element hold.

Shells (2.5-D element)

Shells are essentially 2-D elements that represent 3-D space, thus the term 2.5-D. Shells are excellent for thin 3-D structures, such as body panels, sheet metal, injection molded plastic or any part that can be described as having a thickness that is small relative to its global dimensions. Deflections are given at the nodes, but stresses can be found at the upper and lower surfaces as well as at the midplane. This gives the analyst the ability to extract membrane effects versus bending effects in the results.

Solids (3-D elements)

Solid elements are generally used for 3-D structures not fitting into the shell description. Castings, forgings, blocky structures, and volumes are all good examples of 3-D solid element structures. Solid elements have the benefit of eliminating many assumptions found in the other element types but are generally more difficult to model.

4.12 Mesh Density and Solution Convergence

Mesh density and solution convergence are closely related and the factors which determine that relationship can be controversial. In an effort to meet specific time and accuracy requirements, trade-offs involving modeling time, accuracy, computation time, and cost must be made.

The correct mesh, from a numerical accuracy standpoint, is one that yields no significant differences in the results when a mesh refinement is introduced. Although this concept may sound simplistic, many factors must be considered. Mesh refinements must accurately represent the problem in question if they are to be used in the analysis. Mesh refinements by simple splitting of elements can be misleading unless the newly created nodes conform closely to the original geometry. As refinement progresses, the original element selection must retain its significance. For example, a shell model can be refined to the point that it loses its validity in the area of interest, creating a need for a solid element model.

Determining a mesh density is facilitated by following a few basic guidelines.

Geometric Detail Required

Determine the smallest geometric detail(s) that must be captured in the model to obtain the results. A very sharp radius may cause a stress concentration, but at the same time, it may not be in a load bearing component of the assembly. The modeling required to capture this detail may require a separate local analysis after an analysis of the overall structure or component has been conducted.
Design Detail Available

Observe the degree of detail in the available design data. If the design data is preliminary or incomplete, or if you are using finite element analysis to help define the design, it is best to keep models simple. Take care not to oversimplify models to the point that factors under investigation are missed.

If the design is considered complete and a final verification is being conducted, include as much detail and mesh refinement (including re-mesh iterations for accuracy checks) as time permits.

Comparisons to Previous Work

If you will be comparing your work to other analysis results for the same or similar components, consider using a previously used mesh density which is similar. Consider correlations established with the testing of past models but be prepared to identify improper boundary conditions or load applications, poor modeling techniques, or inadequate mesh density. If such testing flaws are discovered, establish a new standard. Do not accept the work of others until it is fully understood.

Expected Deformed Shape

Determine possible deflection shapes and the mesh densities required to capture them. Estimate the maximum deflection areas and areas of curvature inflection. Observe whether nodal density follows the deflection pattern closely. Note the pattern shown below.

Checks of Convergence

If you plan to do a convergence check, consider performing at least one refinement of the model after the first run. If neighboring elements display large differences in stress, the gradient was probably not captured in these areas, therefore some refinement is recommended.

New techniques for automatically computing convergence by several criteria are available on certain codes or can be customized by the user. Some computer codes will also automatically re-mesh.
nonconverged portions of a model. These techniques are dependent on the load cases and will provide different meshes for different load cases.

The p-version accomplishes refinement by numerically increasing the complexity of each element or nonconverged elements on subsequent re-runs. All automated techniques require that the model be set up to “near perfect” or the convergence may focus on small details that are irrelevant to the specific problem or have not been modeled with elements, boundary conditions or loads that have a natural converged solution. It is best to set up a problem so that there is a chance for convergence even if you do not intend to perform a convergence test. Below is an example of a flat structure that will converge if modeled in shells but will not converge if modeled “more accurately” with solids. Since the shells do not sense stress in the direction of the load, they will converge to the shell theory solution. The solids will attempt to resolve the “point load” and the stresses will go higher and higher as the elements under the load decrease in size.

**Deflection or Stress**

In most cases, far fewer elements are necessary if only deflection or stiffness information is required. Even fewer elements can be used if only the deflection under the load is to be studied. For instance, one beam or one shell can estimate the deflection of a cantilever beam of rectangular section. It may, however, take dozens of shell or solid element to capture the stress at the boundary in a simple cantilever beam. If there is a high gradient of stress over a large area, for example, a web in the corner of a frame, many elements may be required to get the proper deflection.
Areas of Concern

Determine the areas of concern. If you have a known issue in the lab or the field you will certainly want some detail in these areas. If time is limited, you may want to focus only in these areas and use rough approximations elsewhere. If you have no prior knowledge of the component of system and a history search has not turned up any clues, you may have to make a very uniform mesh and refine areas that respond to the applied loads. Use your engineering sense to predict the areas that are prone to trouble. Some typical areas of concern are listed below:

- Tight radii
- Points of load application
- Boundary points
- Attachment points
- Narrow sections
- Abrupt change in section

Element Types

Mesh density is very dependent on the selection of element types therefore select the element type that is appropriate to solve your problem. Determine whether the structure should be considered as a shell type or if a full 3-D model is needed to capture either the state of stress, geometry, or local deflections.

For example, engine blocks can be modeled with shells and beams when stiffness or dynamics is the only consideration. However, the stresses cannot be studied since this component is a 3-D solid when stress becomes a consideration. The time difference to model an engine in solids versus shells and beams can be enormous. Below it is a list of Rules of Thumb regarding element selection:

- Do NOT mix trias and quads whenever possible and if required place trias in areas of non-
Use all trias for back-to-back comparisons when you have determined trias are adequate to
your solution - don’t compare a quad model with a refined tria model.

Use elements of consistent size whenever possible

Do NOT combine shells and solids or beams and shells/solids without first understanding all
the assumptions and implications to your solution

Model solid parts with solid elements

Model thin plate structures with shell elements (thickness < 10-20 x Edge length)

Understand all assumptions for the element type you have selected

Do NOT use degenerated elements unless required

Do NOT mix tetrahedral elements with other element types and, if required, place outside
areas of concern

Use tetrahedral elements when the effort required to model hexahedral elements is
excessive (This can run into weeks of effort versus days for tetras)

Use beam elements when shells or solids require excessive modeling time and effort

Beam elements can be used effectively in beam-like structures or for fasteners and connections.
However, due to the complexity of employing beam elements, they are probably the most misused
elements in the family of elements. The following considerations make using beam elements a
difficult process to perform successfully.

- Shear center
- Warping constraint
- Length to depth ratio
- Shear deflection
- Complex state of stress near end conditions
- Visualizing both the input and the output of beam analysis

Stress results for complex beam sections are only good for global type values computed by classical
beam methods. Beams of circular cross section are an exception because they are easily modeled
and the classical beam results usually match actual performance except at localized end conditions.

Decades of study and research have enabled experts to construct intricate geometry with beam
elements (usually to save computer time) but few individuals understand the complexity of these
techniques. Therefore be prepared to encounter a very involved situation if you choose to employ
this element type.

Timing

Available time is a major determinant in a decision concerning mesh density. A compromise
is usually reached between the amounts of time dedicated to human time and that devoted to
computer time. It is often less time consuming to build a large model by using an automatic mesher,
and increased computer capacity has lessened the need to use special techniques to minimize
the number of elements. Therefore human time has become more focused on productivity and
competitiveness in the market. Pick modeling techniques and elements that provide the most
efficient blend of factors while simultaneously considering fixed external factors. Once you are
convinced that your decisions are solidly supported, discuss your approach with others before
implementing your plan.

Another important factor in the decision on model complexity is the tendency for models to “live”
for an extended period of time. If you feel that the model may have a long life then work at ways to simply the modification and/or redefinition to reduce future effort. This is an often overlooked problem and can cause significant time loss.

**Mesh Density and Solution Convergence Summary**

1. Identify the smallest details that must be captured.

1. Evaluate the level of design detail available.

2. Check for current standards and previous work that will be used for comparison.

3. Estimate the deformed shape and its requirements on your model.

4. Plan for a convergence iteration(s).

5. Select a more detailed modeling approach when stress is important. Consider a two-phase approach to stress solutions in which you make a second model for detailed work to reduce the scope and complexity of your solution.

6. Identify known and predicted areas of concern.

7. Select appropriate element type(s). Mix element types only with caution.

8. Consider timing in your approach.